

CFD SIMULATION USING FLUENT TO DETERMINE THE HEAT TRANSFER
COEFFICIENT OF A PACKED BED SYSTEM

LIM SING WEE

Report submitted in partial fulfillment for the award of the Degree of Bachelor in
Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITY MALAYSIA PAHANG

JUNE 2012

ABSTRACT

In this research, a packed bed is used in the heat exchanger will be considered. Particle to fluid heat transfer coefficient is a primal important when analyzing the performance of a heat exchanger. Basically, a bed packed consists of 44- spherical aluminium particles with proper arrangements is located inside a pipe wall will be considered in this research. A hot fluid is flowing through the packed bed in a solid pipe wall and from here, there will be a pressure drop though this system regarding to the fluid mechanic mechanism. Besides, temperature difference between fluid and solid will cause the convection process and from here heat transfer coefficient can be determined. After the results are taken from the numerical experiment, a comparison between the experimental results with past researcher's results can be done and together with the comparison of calculation using relevant formula.

ABSTRAK

Dalam kajian ini, penukar haba yang dilengkapi dengan system pembungkusan bahan akan dikaji. Zarah dengan pekali pemindahan haba bendalir adalah sangat penting apabila menganalisis prestasi penukar haba. Secara umumnya, system pembungkusan bahan dalam paip ini mengandungi 44- bebuli aluminum mengikut susunan yang betul akan dikaji. Cecair panas akan melalui system pembungkusan bahan ini dan dari sini akan berlaku perubahan tekanan dalam system ini mengikut mekanisma mekanik bendalir. Sementara itu, perbezaan suhu pada pepejal dan cecair dalam system ini akan menyebabkan proses olakan dan dari sini pekali pemindahan haba boleh ditentukan. Selepas keputusan diambil dari eksperimen, perbandingan di antara keputusan eksperimen dengan keputusan penyelidikan lepas boleh dilakukan dan bersama-sama dengan perbandingan pengiraan menggunakan formula yang berkaitan.

TABLE OF CONTENTS

TITLE	PAGE
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENT	viii
LIST OF TABLES	x
LIST OF FIGURES	xi
LIST OF SYMBOLS	xiii

CHAPTER 1 INTRODUCTION

1.1	Background of study	1
1.2	Problem Statement	2
1.3	Objective	3
1.4	Scopes	3

CHAPTER 2 LITERATURE REVIEW

2.1	Introductions	4
2.2	Bed packed system	4
2.2.1	Relevant studies	5
2.2.2	Pressure drop along the bed packed	10
2.2.3	Pressure drop of bed packed in different fluid flow velocity	11
2.2.4	Heat transfer coefficient in different fluid flow velocity	11
2.3	Computational Fluid Dynamic (CFD)	12
2.3.1	History of CFD	12

CHAPTER 3 RESEARCH METHODOLOGY

3.1	Introductions	14
3.2	Flow chart	14
3.2.1	Overall research methodology	14
3.2.2	Steps if CFD analysis	15
3.3	Numerical Experiment Setup	16
3.3.1	Geometrical Modelling	16

3.3.2	Design Modular	18
3.3.3	Mesh	18
3.3.4	Fluent	19
3.4	Apparatus	22

CHAPTER 4 RESULTS AND DISCUSSIONS

4.1	Introductions	23
4.2	Numerical experiment results	23
4.2.1	Air	23
4.2.2	Water	25
4.2.3	Nanofluid: Aluminium Oxide + Water	26
4.3	Calculation results	28
4.3.1	Calculation for air	28
4.3.2	Calculation for water	29
4.3.3	Different between calculated values and the simulation result values	29
4.4	Comparison of numerical experiment with relevant research	30

CHAPTER 5 CONCLUSION AND RECCOMENDATIONS

5.1	Conclusions	33
5.2	Recommendation	34

REFERENCES 35

APPENDIX A: Sample of calculation 37

APPENDIX B: Simulations Results 40

LIST OF TABLES

TABLE NO.	TITLE	PAGE
1	Boundary condition for carbon dioxide	6
2	Boundary condition for air and carbon dioxide	8
3	Boundary conditions for water as fluid properties	21
4	Simulation results for air	23
5	Simulation results for water	25
6	Simulation results for different concentration of aluminium oxide in water	26
7	Calculated Nusselt Number for air	28
8	Calculated pressure drop for water	29
9	Nanomaterial properties	30
10	Water properties	30
11	Different concentration of aluminium oxide in water properties	30

LIST OF FIGURES

FIGURE NO.	TITLE	PAGE
1	Graph Nusselt versus Reynold number for carbon dioxide	7
2	Nusselt number versus Reynold Number of air and carbon dioxide	9
3	Convective particle to fluid heat transfer	10
4	Overall process of research	14
5	Steps of CFD simulations	15
6	Front view of bed packed model	17
7	Solidwork isometric view of bed packed model	17
8	46 parts in Design Modular of bed packed model	18
9	Meshing medium size bed packed model	19
10	Graph Pressure Drop versus Reynold Number for air	24
11	Graph Nusselt Number versus Reynold Number for air	24
12	Graph Pressure Drop versus Reynold Number for water	25
13	Graph Nusselt Number versus Reynold Number for water	26
14	Graph Pressure Drop versus Reynold number of different concentration of aluminium oxide in water	27
15	Graph Nusselt Number versus Reynold number for different concentration aluminium oxide in water	28
16	Graph Nusselt Number versus Reynold Number for different temperature conducted	31
17	Graph Nusselt Number versus Reynold Number of different concentration of Aluminium Oxide and different temperature conducted	32
18	Temperature contour for water at $Re = 2514$	40
19	Velocity vector of water at $Re = 2514$	40
20	Velocity vector of water at $Re = 2514$, near the wall surface.	41
21	Wake region for water at $Re = 1235$	41
22	Temperature contour for air at $Re = 153.16$	42
23	Velocity vector for air at $Re = 110.13$, near the wall surface	42

24	Temperature contour for 2% concentration of aluminium oxide in water at $Re = 2658$	43
25	Temperature contour for 4% concentration of aluminium oxide in water at $Re = 4505$	43

LIST OF SYMBOLS

D = effective particle diameter

s_v = specific surface of a particle

S_p = surface area of particle

V_p = volume of particle

ρ = fluid density

ε = dimensionless void fraction

μ = fluid viscosity

f_p = friction factor

Re = Reynold Number

Nu = Nusselt Number

St = Stranton number

C_7H_8 = toluene

Al_2O_3 = aluminium oxide

H = height of bed

ΔP = pressure drop

v = velocity of fluid

C_{nf} = specific heat of nanofluid

ρ_{nf} = density of nanofluid

K_{nf} = thermal conductivity of nanofluid

μ_{nf} = viscosity of nanofluid

Φ = percentage of concentration of nanomaterial in nanofluid

φ = amount of concentration nanomaterial in nanofluid (in percentage)

CHAPTER 1

INTRODUCTIONS

1.1 BACKGROUND OF STUDY

The convective coefficients of a packed bed heat exchanger are important in many process heat transfer equipment. As an example, within a heat exchanger the evaluation of temperature profile as well as the heat transfer rates of the bed packed is essential to control the performance of the heat exchanger. Hence, in this study is required to develop a packed bed heat exchanger model with suitable software for the estimation of heat transfer coefficients using water as the working fluid. Fluid may be heated from the wall while flowing through the packed bed system.

From here, by applying the theory of heat flow or the movement of thermal energy from place to place, heat is transferred in three methods that are conduction, convection and radiation. Conduction is heat transfer requires the physical contact of two objects. In the case of a wall, heat is conducted through the layers within the wall from the warmer side to the cooler side. Meanwhile, convection is heat transfer due to fluid or air flow. In here, heat is transferred from the wall of water is called as convection. For radiation, heat is transferred when surfaces exchange electromagnetic waves, such as light, infrared radiation, UV radiation or microwaves. Although radiation does not require any fluid medium or contact, but does require an air gap or other transparent medium between the surfaces exchanging radiation.

In such a packed bed operated under steady-state conditions, a difference in local temperature between the fluid and the particle may exist, but the overall solid and fluid temperature profiles are considered to be identical to each other. The temperature profiles in the bed are then predicted in terms of effective thermal conductivities and

wall heat transfer coefficients. An extensive review of the aforementioned can be found in Wakao and Kuguei, 1982.

A Computerized Fluid Dynamic (CFD) simulation is a most suitable strategy for the estimation of effective thermal conductivities as well as wall heat transfer coefficients. CFD is a tool uses numerical methods and algorithms to analyze systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. The simulation of CFD is performed using the FLUENT software. CFD allows us to obtain a more accurate view of the fluid flow and heat transfer mechanisms present in packed bed heat exchanger.

Fluent is a computer program for modeling fluid flow and heat transfer in complex geometries. Fluent provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Fluent also allows you to refine your grid based on the flow solution.

1.2 PROBLEM STATEMENT

The problems begin with a hot fluid is flowing through a hollow tube pipe with a packing material inside the pipe. The packing materials are spherical solid materials. The fluid is flowing through each spherical packing material through the column of the packing material. The energy of the hot fluid is transferred to the solid sphere through the convection process. The differences of temperature of the pipe wall and the fluid also make the energy transfer of fluid to the wall. Those energy transfers are simulated using the Fluent software in computational fluid dynamic (CFD). Although the simulated results are not as accurate as physical experiment results but simulated results are almost can be referred results. CFD simulations are relatively inexpensive because the cost of the powerful computer to simulate the design can be cheaper than the experimental solution. CFD simulations can be executed in a short period of time. Hence, the fastest and almost an accurate way to solve that problem or through the computer software and using CFD simulation.

1.3 OBJECTIVE

To determine convective heat transfer coefficient and pressure drop of a packed bed heat exchanger using Fluent software.

1.4 SCOPES

1. Develop the model of a packed bed system in Solidwork or any other commercial software available compatible with CFD package
2. Import the model and initialize for boundary conditions. Process/ Execute the model using the properties of water
3. The experimental data available for the water is to be validated with computational results for heat transfer coefficients
4. Evaluate the numerical heat transfer coefficient for nanofluid using the properties developed in the form of equations.

CHAPTER 2

LITERATURE REVIEWS

2.1 INTRODUCTION

The aim of this chapter is to provide the past research about the bed packed system and computational fluid dynamics (CFD) analysis in three dimension model. In order to understand more on this research and to achieve the objective of the research, reviewing back past research studies are needed to provide more useful information and point.

2.2 BED PACKED SYSTEM

Bed packed system is a hollow tube filled with fixed layer of small particles or packing material. The packing material can be any sizes and shapes but for this research, it is spherical aluminum particles and a fluid is flowing through the bed packed particles. The purpose of this system is to use for processes involving absorption, absorption of a solute, distillation, filtration and separation (Geankoplis, p.125). One of the studies in this research is pressure drop. It is because pressure drop is important to determine the energy requirement to pump a fluid at any bed packed system. Besides, from the viewpoint of fluid dynamics, the most important cases are the pressure drop required for fluid to flow through the column at a specified flow rate.

In order to calculate this amount, we are dependent on the correlation of coefficient friction due to the Ergun. Ergun relates the flows and pressure drops to a Reynolds number and friction factor respectively. The Reynolds number for packed

beds, Re_p , depends upon the controlled variable U_{bs} and the system parameters ρ , ε , μ , and D and is defined as (Bird et al., 1996):

$$Re_p = \frac{D \times U_{bs} \times \rho}{\mu (1 - \varepsilon)}$$

where, D is effective particle diameter $= \frac{6}{s_v}$, s_v is specific surface of a particle $= S_p / V_p$,

S_p is surface area of particle and V_p is volume of particle. ρ is the fluid density, ε is the dimensionless void fraction defined as the volume of void space over the total volume of packing, and μ is the fluid viscosity.

The friction factor, f_p , in the Ergun equations for Reynolds's number range between 1 and 2500 are:

$$f_p = \frac{150}{Re_p} + 1.75$$

2.2.1` Relevant Studies

In “CFD studies on particle-to-fluid mass and heat transfer in packed beds: Free convection effects in supercritical fluids” by Guardo. A, Coussirat. M, Recasens. F, Larrayoz. M. A, Escaler. X have checked CFD capabilities for predicting particle-to-fluid mass/heat transfer coefficients when a supercritical fluid was used as a solvent in a packed bed reactor. First, numerical simulations are presented for validation model cases (forced convection at low pressure). Numerical simulation is done to model the mass transfer of mixed convection under high pressure, the analysis results was obtained was compared with experimental data previously issued by (Stüber et al., 1996), and to the heat transfer analogy proposed by (Guardo et al., 2006). Numerical results obtained presented in this study, to validate the idea that the modified correlation presented by (Guardo et al., 2006) can be used to describe the phenomenon of heat transfer in packed bed under mixed convection regime at high pressures. The boundary conditions in this study are as follow

Table 1: Boundary condition for carbon dioxide

Boundary condition	Low pressure	High pressure
Mass transfer simulations		
Circulating fluid		CO_2
C_7H_8 concentration at inlet (mol/m^3)		0
C_7H_8 concentration at particle surface (equilibria) (mol/m^3)	5.95	120–190
Pressure (Pa)	101325	$9-9.2 \times 10^6$
Mass flow at the inlet	–	0.015–0.100
Velocity at the inlet (m/s)	7.5×10^{-4} – 7.5×10^{-1}	–
Heat transfer simulations		
Circulating fluid	Air	CO_2
Temperature at the inlet (K)	298	330
Temperature at particle surface (K)	423	340
Pressure (Pa)	101325	1×10^7
Mass flow at the inlet	–	0.013–0.132
Velocity at the inlet (m/s)	3×10^{-4} – 7.5×10^{-1}	–

The fluid was taken to be incompressible, Newtonian, and in a laminar or turbulent flow regime. CO_2 , air and toluene at standard conditions were chosen as the simulation fluids. Incompressible ideal gas law for density and viscosity were applied to the model for the production of these variables depends on temperature. For the high-pressure simulations, the fluid was taken to Newtonian, in laminar flow regime and with variable density. CO_2 and toluene, property that has been incorporated into the solver code using the (UDE) user-defined functions and user-defined formula (UDF) was chosen as a fluid simulation in this case, under high pressure.

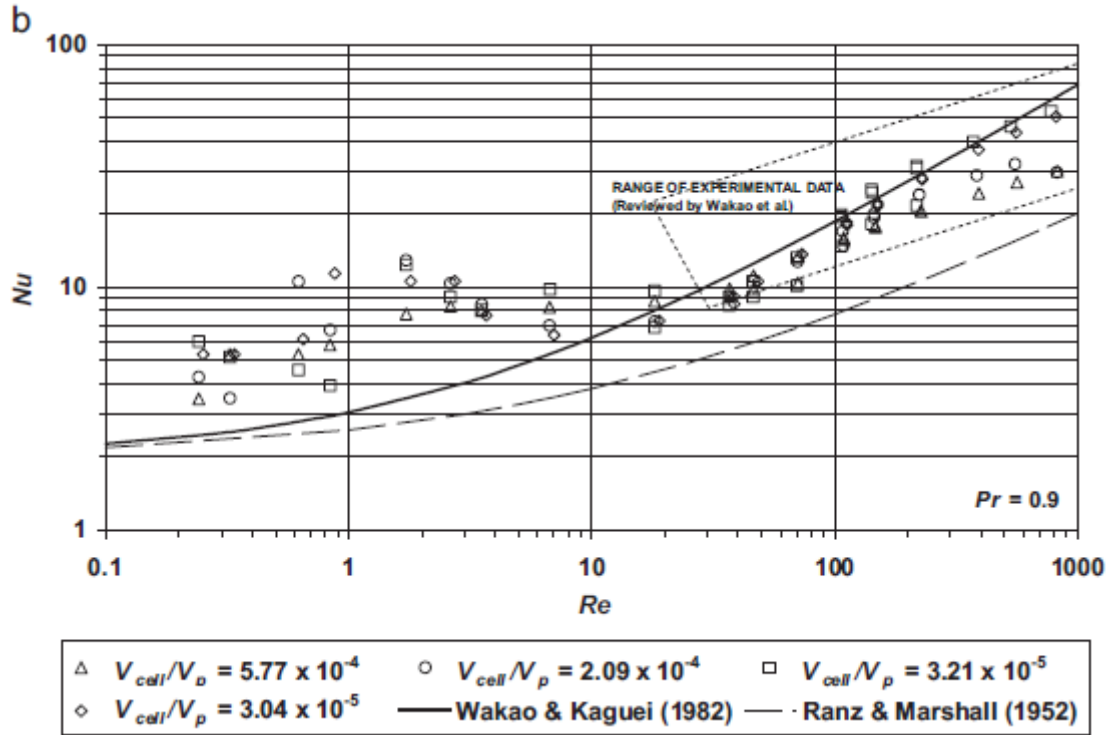


Figure 1: Graph Nusselt vs Reynold number for carbon dioxide

In the end, they find that at lower Reynolds numbers ($Re < 10$), mass and heat transfer results obtained show that the fitting against Wakao's correlation is not good because of the CFD model in predicting Nu at low Re has stated by (Guardo et al., 2006). There is a discrepancy between the results for meshes tested in heat and mass transfer simulations for $Re > 300$, due to the fact that obtained at higher Re , turbulent transport term in the transport equation is important. Therefore, it is requires a denser mesh around the particles surface to capture the turbulence phenomena involved and the associated improvement in the boundary layer mixing (Guardo et al., 2006). The results obtained with the finer meshes fit better the prediction of (Wakao and Kaguei, 1982) in the turbulent flow zone ($Re > 300$) related to a better determination of the vorticity energetic scales effects.

In “CFD study on particle-to-fluid heat transfer in fixed bed reactors: Convective heat transfer at low and high pressure” by Guardo. A, Coussirat. M, Recasens. F, Larrayoz. M.A, Escaler. X have review CFD capabilities for predicting particle-to-fluid heat transfer coefficients when a supercritical fluid is used as a solvent in a fixed bed reactor. At first, numerical simulations are presented for a validation

model case (single sphere model), and next is presenting convective particle-to-fluid heat transfer at low pressure. The results obtained are used to analyze mesh dependence of the numerical results at low flow velocities. Finally, mixed convection at high pressure is modeled and analyzed. Numerical results obtained are compared to accepted correlations and a CFD-based correlation for particle-to-fluid heat transfer at high pressure is presented. The boundary conditions are;

Table 2: Boundary condition for air and carbon dioxide

Boundary condition	Low pressure	High pressure
Circulating fluid	Air	CO_2
Temperature at the inlet, K	298	330
Temperature at packing surface, K	423	340
Pressure, Pa	101 325	1×10^7
Mass flow at the inlet, kg/m^2s	—	0.013–0.132
Velocity at the inlet, m/s	3×10^{-4} – 7.5×10^{-1}	—

Model consists of a single sphere suspended in a box. In the CFD model the infinite fluid was limited in a box with a square flow inlet plane of seven sphere diameters and a length of 16 sphere diameters to keep the model reasonable in size (Nijemeisland, 2000). To discard the presence of wall effects on temperature and velocity profiles, models with flow inlet planes with sizes of 2, 3, 4, 6, 8 and 9 diameters were created. An unstructured tetrahedral mesh is built in the fluid region. No mesh is built in the sphere interior. The sphere in the box was designed with the same dimensions as the spheres used in the fixed bed model.

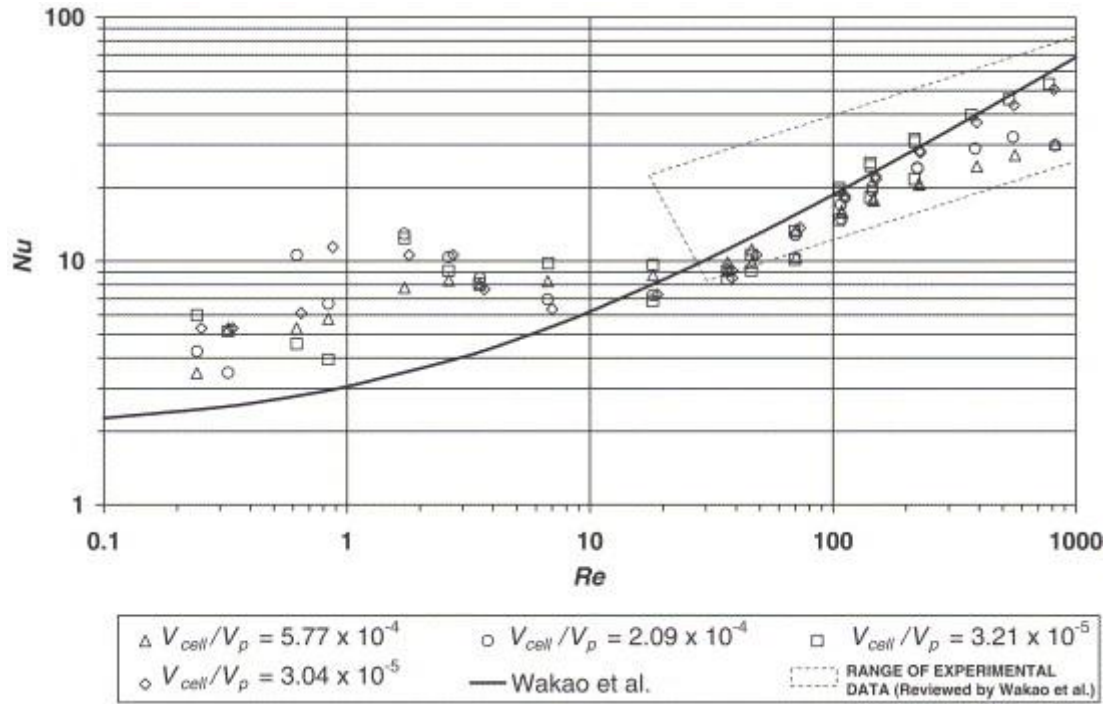


Figure 2: Nusselt number versus Reynold Number for air and carbon dioxide

From the results, they notice that in the laminar flow and transition zone ($Re < 300$), the results not depending on the mesh density. At lower Reynolds numbers ($Re < 10$) shows the results that the fitting against (Wakao, 1976) the correlation is not good. For a single velocity condition different meshes give results in a wide range of Nu and no relationship with mesh density can be determined.

There is a divergence between the results obtained for tested meshes for higher values of Re , due to the fact at higher Re , turbulent transport term in the transport equation need to consider. An accurate turbulence modeling requires a denser mesh around the particles surface in order to capture in a more suitable way the involved turbulence phenomena in the boundary layer (Guardo et al., 2005). A divergence in the results obtained with the low density meshes and the high density meshes can be seen for $Re > 300$.

In “Heat and Flow Characteristics of Packed Bed by (Achenbach. E, 1995), mass transfer experiments with single spheres are preferably conducted according to the method of naphthalene sublimation in air. The majority of the present experiments were conducted using a bed diameter of $D = 0.983$ m and a bed height of $H = 0.84$ m. To eliminate wall effects, the core wall was structured such that a regular orientation of the

spheres adjacent to the wall was avoided. The sphere diameter was $d = 0.06$ m. The void fraction was experimentally determined to be 0.387. The heat transfer experiments were carried out by applying the method of the electrically heated single sphere in an unheated packing. The test spheres were manufactured from copper, the surface being highly polished and covered with a silver layer to keep the contribution of thermal radiation low.

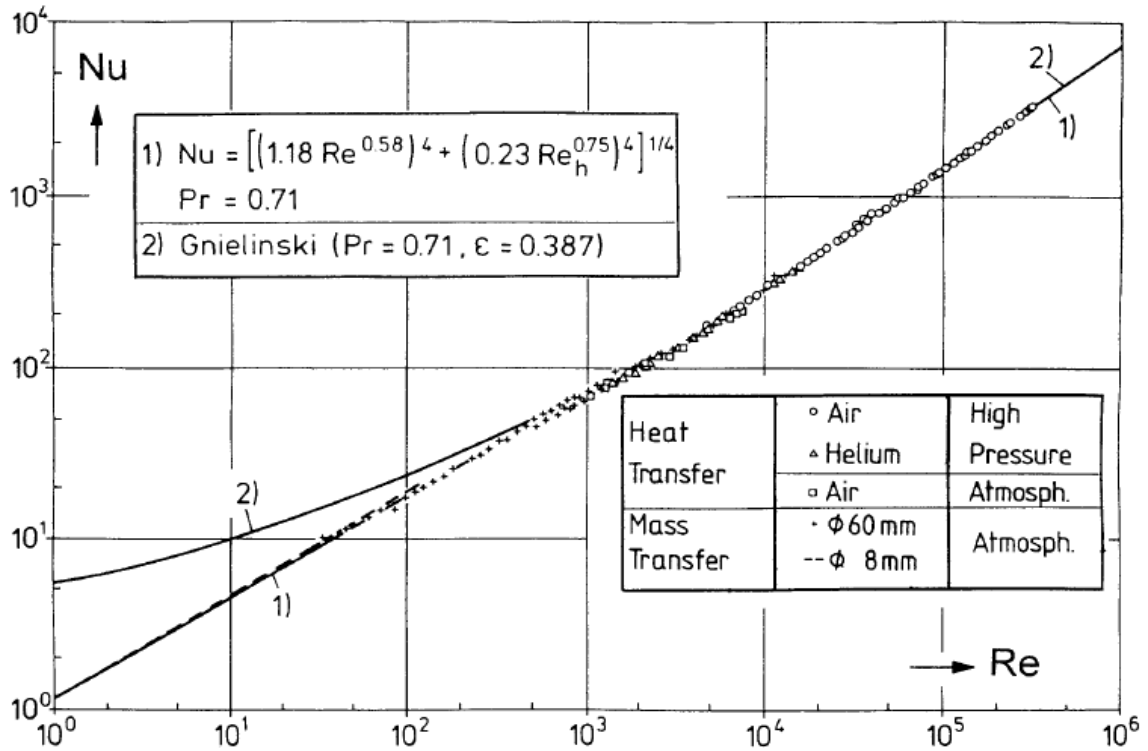


Figure 3: Convective particle to fluid heat transfer

2.2.2 Pressure drop along the bed packed

The increasing of pressure drop along the bed pack is due to the wake region is created by each solid particle in the bed packed when water flow pass through it. Wake region is a recirculating flow immediately behind a moving or stationary solid body and it is caused by the flow of surrounding fluid around the body. The pressure is a maximum at the stagnation point (first point contact of fluid and spherical particles) and gradually decreases along the front half of the spherical particles of bed packed. The pressure starts to increase in the rear half of the spherical particles of bed packed and the particle now experiences an adverse pressure gradient. Consequently, the flow

separates from the surface and creating a highly turbulent region behind the spherical particles of bed packed called the wake region. The pressure inside the wake region remains low as the flow separates and a net pressure force (pressure drag) is produced.

2.2.3 Pressure drop of bed packed in different fluid flow velocity

The increasing pressure drop in different velocity is because of increasing of velocity is directly proportional to increasing of pressure drop value in pressure drop formula.

Pressure drop can be calculated using the following formula:

$$\Delta P = \frac{v^2 \times f \times L \times \rho}{2D_{pipe}}$$

Where;

ΔP = pressure drop in Pascals (Pa)

v = velocity in metres per second (m/sec)

f = friction factor

L = length of pipe in metres (m)

ρ = density of the fluid in kilograms per cubic metre

D_{pipe} = inside diameter of pipe in metres (m)

Hence, when the velocity of the fluid is increasing, the pressure drop values are also increasing.

2.2.4 Heat transfer coefficient in different fluid flow velocity

All the graph of Nusselt number versus Reynold number in different fluid properties must shows the increasing pattern. It is because due to the convection process between fluid and the bed packed. Convection is the mode of energy transfer between solid surface and the adjacent fluid in motion and it involves combined effect of conduction and fluid motion. Hence, the faster the fluid motion, the greater the convection heat transfer and the higher the Nusselt number will be.

2.3 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations. CFD provides numerical approximation to the equations that govern fluid motion. These equations are then discretized to produce a numerical analogue of the equations.

All CFD codes contain three main elements:

1. A pre-processor, which is used to input the problem geometry, generate the grid, define the flow parameter and the boundary conditions to the code.
2. A flow solver, which is used to solve the governing equations of the flow subject to the conditions provided. There are four different methods used as a flow solver: (i) finite difference method; (ii) finite element method, (iii) finite volume method, and (iv) spectral method.
3. A post-processor, which is used to massage the data and show the results in graphical and easy to read format.

2.3.1 History of CFD

In England, Lewis Fry Richardson (1881-1953) developed the first numerical weather prediction system CFD approximation in 1922 when he divided physical space into grid cells and used the finite difference approximations of Bjerknes's "primitive differential equations". His own attempt to calculate weather for a single eight-hour period took six weeks and ended in failure. His model's enormous calculation requirements led Richardson to propose a solution he called the "forecast-factory". The "factory" would have involved filling a vast stadium with 64,000 people. Each one, armed with a mechanical calculator, would perform part of the flow calculation. A leader in the centre, using coloured signal lights and telegraph communication, would coordinate the forecast. What he was proposing would have been a very rudimentary CFD calculation. The earliest numerical solution for flow pass a cylinder is in 1933 that is by Thom. A, publishing 'The Flow Past Circular Cylinders at Low Speeds', Proc. Royal Society, A141, pp. 651-666, London, 1933.

During the 1960s, the theoretical division of NASA at Los Alamos in the U.S. contributed many numerical methods that are still in use in CFD today, such as the following methods: Particle-In-Cell (PIC), Marker-and-Cell (MAC), Vorticity- Stream function methods, Arbitrary Lagrangian-Eulerian (ALE) methods, and the ubiquitous $k - \epsilon$ turbulence model. In the 1970s, a group working under D. Brian Spalding, at Imperial College, London, developed Parabolic flow codes (GENMIX), Vorticity-Stream function based codes, the SIMPLE algorithm and the TEACH code, as well as the form of the $k - \epsilon$ equations that are used today (Spalding & Launder, 1972). They went on to develop Upwind differencing, 'Eddy break-up' and 'presumed PDF' combustion models. Another event of CFD industry was in 1980 when Suhas V. Patankar published "Numerical Heat Transfer and Fluid Flow", probably the most influential book on CFD to date, and the one that spawned a thousand CFD codes.

It was in the early 1980s that commercial CFD codes came into the open market place in a big way. The use of commercial CFD software started to become accepted by major companies around the world rather than their continuing to develop in-house CFD codes. Commercial CFD software is therefore based on sets of very complex non-linear mathematical expressions that define the fundamental equations of fluid flow, heat and materials transport. These equations are solved iteratively using complex computer algorithms embedded within CFD software. The net effect of such software is to allow the user to computationally model any flow field provided the geometry of the object being modelled is known, the physics and chemistry are identified, and some initial flow conditions are prescribed.

CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modeling fluid flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTIONS

Basically, the flow of this research is as shown as figure 1. When there is an error in this simulation, the research has to go backward to any cases before the simulation takes place. In this research, geometry model of this bed packed is looked simple but it requires lots of steps to build it. From the overall, a proper geometrical model will define a good result in simulation.

3.2 FLOW CHART

3.2.1 Overall Research Methodology

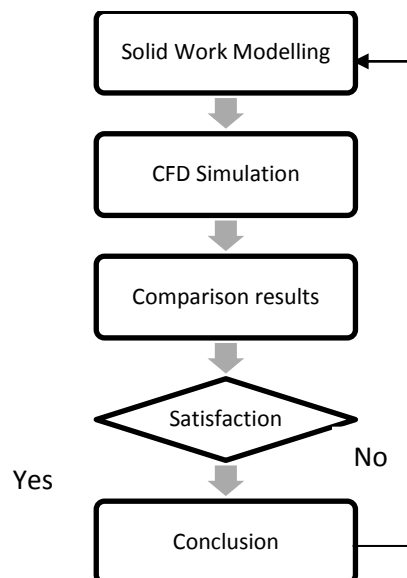


Figure 4: Overall process of research

3.2.2 Steps of CFD analysis

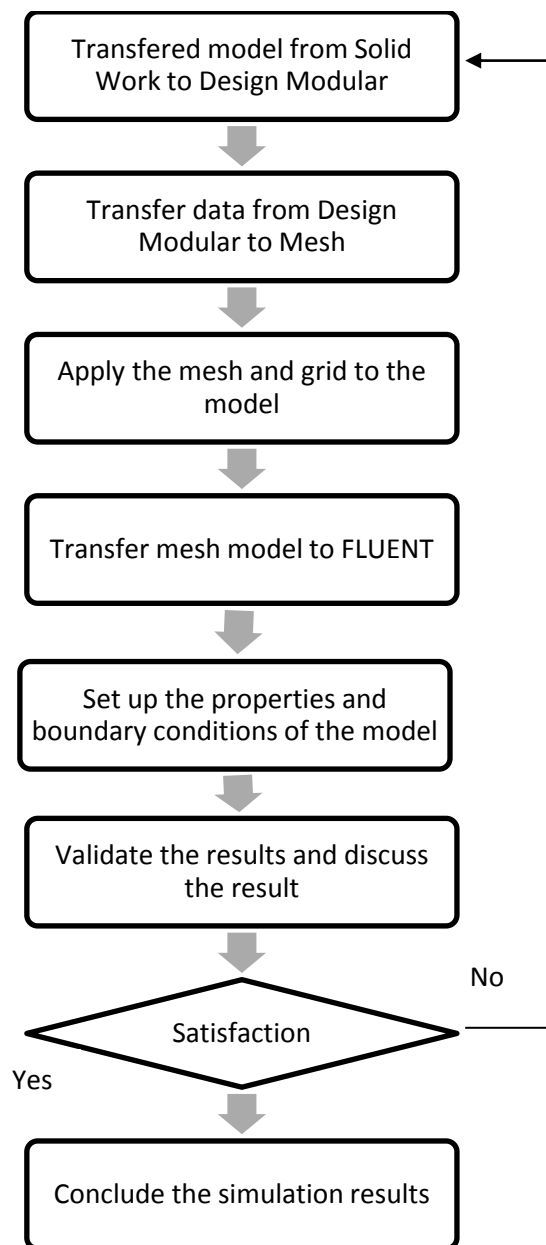


Figure 5: Steps of CFD simulations